PROBABILISTIC FINITE ELEMENT ANALYSIS OF STEEL ROOF TRUSS USING ANSYS

KHONG JIA WEI

A report submitted in partial fulfillment of the
Requirement for the award of the
Degree of Bachelor of Civil Engineering

FACULTY OF CIVIL ENGINEERING AND EARTH RESOUCES
UNIVERSITY MALAYSIA PAHANG

JUNE 2012

ABSTRACT

Truss is one of the important components for a structure and need to be designed in such a way that they have enough strength and rigidity to satisfy the strength and serviceability limitation. So, truss required a proper method to be analyzes. Among all those methods, finite element analysis shall be one of the best methods in analyzing the truss because it can analyze complex structure without simplify the structure. However, during the designation of truss, there is a possibility that randomness and uncertainty may occur, thus probabilistic analysis shall be apply to solve this problem. Here, the study was conducted to prove that a steel roof truss under the existence of randomness and uncertainty can be analyzed by apply probabilistic finite element analysis using ANSYS. This report presents an alternative way in the designation of steel roof truss with many uncertainties.

ABSTRAK

Kekuda merupakan salah satu daripada bahagian utama dalam sesuatu struktur dan perlu direkabentuk supaya mencapai kekuatan dan kekukuhan yang mencukupi untuk memenuhi had kekuatan dan had kebolehkhidmatan. Oleh itu, kekuda memerlukan satu cara yang sesuai untuk dianalisis. Di kalangan cara-cara tersebut, analisis unsur terhingga merupakan salah satu cara terbaik antara cara-cara untuk menganalisis kekuda sebab cara ini boleh menganalisis struktur kompleks tanpa memudahkan struktur tersebut. Walau bagaimanapun, masih ada kemungkinan bahawa kerawakan dan ketidakpastian akan berlaku pada masa rakabentuk kekuda, sebab itu analisis kebarangkalian perlu dijalankan untuk menyelesaikan masalah ini. Di sini, kajian ini dijalankan untuk membuktikan bahawa kekuda keluli bumbung yang mengalami masalah kerawakan dan ketidakpastian boleh dianalisis dengan mengunakan cara analisis unsure terhingga dan analisis kebarangkalian di bawah penolongan perisian computer ANSYS. Laporan ini membentangkan satu singkat dalam rekabentuk kekuda keluli bumbung yang mengalami masalah kerawakan dan ketidakpastian.

TABLE OF CONTENT

CHAPTER	TITLE	PAGE
	DECLARATION	**ii**
	DEDICATION	**iv**
	ACKNOWLEDGEMENT	**v**
	ABSTRACT	**vi**
	ABSTRAK	**vii**
	TABLE OF CONTENTS	**viii**
	LIST OF TABLES	**xi**
	LIST OF FIGURES	**xii**
	LIST OF SYMBOLS	**xv**
1	**INTRODUCTION**	**1**
	1.1 General	1
	1.2 Problem Statement	3
	1.3 Objectives	3
	1.4 Scope of Works	4
2	**LITERATURE REVIEW**	**5**

	2.1 Introduction	5
	2.2 Steel Roof Truss	7
	2.3 Finite Element Analysis	9
	2.4 Probabilistic Analysis	12
3	**METHODOLOGY**	**13**
	3.1 Introduction	13
	3.2 Preprocessing Phase	14
	3.3 Solution Phase	16
	3.4 Postprocessing Phase	18
	3.5 Probabilistic Analysis	18
4	**RESULTS AND DISCUSSIONS**	**22**
	4.1 Introduction	22
	4.2 Deterministic Model Information	23
	4.3 Probabilistic Model Information	24
	4.3.1 Random Input Variables	24
	4.3.2 PDF and CDF of EX	24
	4.3.3 PDF and CDF of POISSON	25
	4.3.4 PDF and CDF of AREA	26
	4.3.5 PDF and CDF of DENSITY	27
	4.3.6 PDF and CDF of UDL	28
	4.3.7 PDF and CDF of WIND	29
	4.3.8 PDF and CDF of FORCE1	30

4.3.9 PDF and CDF of FORCE2 31

4.3.10 PDF and CDF of FORCE3 32

4.4 Probabilistic Analysis Results 33

4.4.1 Statistic of the Probabilistic Results 33

4.4.2 Sample History Plots 35

4.4.3 Histogram Plots 43

4.4.3.1 Histogram for Input Variables 43

4.4.3.2 Histogram for Output Parameter 48

4.4.4 Cumulative Distribution Function Plots 50

4.4.4.1 CDF of Input Variables 50

4.4.4.2 CDF of Output Parameters 55

4.4.5 Sensitivity Plots 57

5 CONCLUSION 61

5.1 Conclusion 61

5.2 Recommendations 62

REFERENCES 63

APPENDIX 65

LIST OF TABLES

TABLE NO.	TITLE	PAGE
1.1	Properties of Steel	2
3.1	Specifications and Properties of Steel Roof Truss	13
3.2	Keypoints of the Steel Roof Truss	14
4.1	Material Properties of the Deterministic Model	23
4.2	Random Input Variable Specifications	24
4.3	Statistic of the Random Input Variables	34
4.4	Statistic of the Random Output Parameters	34

LIST OF FIGURES

FIGURE NO.	TITLE	PAGE
1.1	Howe Truss	2
2.1	Common Shape for Steel	7
3.1	Library of Element Types	15
3.2	Material Model Behaviour	16
3.3	Window to Apply Constraints	17
3.4	Model with Load Applied	17
3.5	Note of Solution Done	18
3.6	Define Random Variable	19
3.7	Random Input Variables	20
3.8	Random Output Variables	20
4.1	Deterministic Model Information	23
4.2	PDF and CDF of Input Random Variable EX	25
4.3	PDF and CDF of Input Random Variable POISSON	26
4.4	PDF and CDF of Input Random Variable AREA	27
4.5	PDF and CDF of Input Random Variable DENSITY	28
4.6	PDF and CDF of Input Random Variable UDL	29
4.7	PDF and CDF of Input Random Variable WIND	30
4.8	PDF and CDF of Input Random Variable FORCE1	31
4.9	PDF and CDF of Input Random Variable FORCE2	32

4.10	PDF and CDF of Input Random Variable FORCE3	33
4.11	Sample Values for Maximum Deflection	35
4.12	Mean Value History for Maximum Deflection	36
4.13	Standard Deviation History for Maximum Deflection	36
4.14	Minimum Value History for Maximum Deflection	37
4.15	Maximum Value History for Maximum Deflection	37
4.16	Sample Values for Maximum Von-mises Stress	38
4.17	Mean Value History for Maximum Von-mises Stress	38
4.18	Standard Deviation History for Maximum Von-mises Stress	39
4.19	Minimum Value History for Maximum Von-mises Stress	39
4.20	Maximum Value History for Maximum Von-mises Stress	40
4.21	Sample Values for Maximum Von-mises Strain	40
4.22	Mean Value History for Maximum Von-mises Strain	41
4.23	Standard Deviation History for Maximum Von-mises Strain	41
4.24	Minimum Value History for Maximum Von-mises Strain	42
4.25	Maximum Value History for Maximum Von-mises Strain	42
4.26	Histogram of Input Variable Modulus of Elasticity	43
4.27	Histogram of Input Variable Poisson Ratio	44
4.28	Histogram of Input Variable Cross-Sectional Area	44
4.29	Histogram of Input Variable Density	45
4.30	Histogram of Input Variable Uniform Distributed Load	45
4.31	Histogram of Input Variable Wind Load	46
4.32	Histogram of Input Variable Point Load 1	46
4.33	Histogram of Input Variable Point Load 2	47
4.34	Histogram of Input Variable Point Load 3	47
4.35	Histogram of Output Parameter Maximum Deflection	48

4.36	Histogram of Output Parameter Maximum Von-mises Stress	49
4.37	Histogram of Output Parameter Maximum Von-mises Strain	49
4.38	CDF of Input Variable Modulus of Elasticity	50
4.39	CDF of Input Variable Poisson Ratio	51
4.40	CDF of Input Variable Cross-Sectional Area	51
4.41	CDF of Input Variable Density	52
4.42	CDF of Input Variable Uniform Distributed Load	52
4.43	CDF of Input Variable Wind Load	53
4.44	CDF of Input Variable Point Load 1	53
4.45	CDF of Input Variable Point Load 2	54
4.46	CDF of Input Variable Point Load 3	54
4.47	CDF of Output Parameter Maximum Deflection	55
4.48	CDF of Output Parameter Maximum Von-Mises Stress	56
4.49	CDF of Output Parameter Maximum Von-Mises Strain	56
4.50	Linear Correlation Sensitivities Plot for Maximum Deflection	57
4.51	Rank-Order Correlation Sensitivities Plot for Maximum Deflection	58
4.52	Linear Correlation Sensitivities Plot for Maximum Von-Mises Stress	58
4.53	Rank-Order Correlation Sensitivities Plot for Maximum Von-Mises Stress	59
4.54	Linear Correlation Sensitivities Plot for Maximum Von-Mises Strain	59
4.55	Rank-Order Correlation Sensitivities Plot for Maximum Von-Mises Strain	60

LIST OF ABBREVIATIONS

TERMS.	MEANING
PDF	Probability Density Function
CDF	Cumulative Distribution Function
EX	Modulus of Elasticity
POISSON	Poisson Ratio
AREA	Cross Sectional Area
DENSITY	Density
UDL	Uniform Distributed Load
WIND	Wind Load
FORCE	Concentrated Load

CHAPTER 1

INTRODUCTION

1.1 General

The finite element analysis is a numerical technique. It was an extension of matrix method of structural analysis. In finite element method, actual component is replaced by a simplified model, identified by a finite number of elements connected at common point called nodes, with an assumed behaviour of each element to the set of applied load, and evaluating the unknown variable such as displacement and temperature at this finite number of points. In this method all the complexities of the problems, like varying shapes, boundary conditions and loads are preserved, but the solution obtained are approximate. Nowadays civil engineer used this method extensively for the analysis of beam, space frames, plates, shells, foundations, etc. Finite element analysis can handled both static and dynamic problems.

In this research, finite element analysis were carried out in assisting with computer software ANSYS. Instead of being popular to be used for finite element analysis packages which can analyses several complex structures, included roof trusses with a complex structure, ANSYS also provide other extension on many engineering field testes such as aerodynamic, electromagnetic and so on. Besides that, ANSYS can carry out thermal analysis for the structure while other software like ESTEEM don't have this function.

Focusing on steel roof trusses, this study was a research effort in the development of a safe and efficient roof truss system especially for residential construction and upgrading. The imposed load for a simple residential building is

$0.75kN/m^2$ since there are only access to the roof for maintainances and repair works. The total dead load, which included self weight of roof tiles, purlins and truss members is assumed as $1.5kN/m^2$. The material properties of steel roof trusses are linear, elastic and isotropic. The properties of steel are as table 1.1 while the shape of the trusses is howe shape as shown in figure 1.1.

Table 1.1 Properties of steel

Properties	Value	Unit
Density	7850	Kg/m^3
Elastic modulus	200	GPa
Shear modulus	77.9	GPa
Poisson's ratio	0.3	

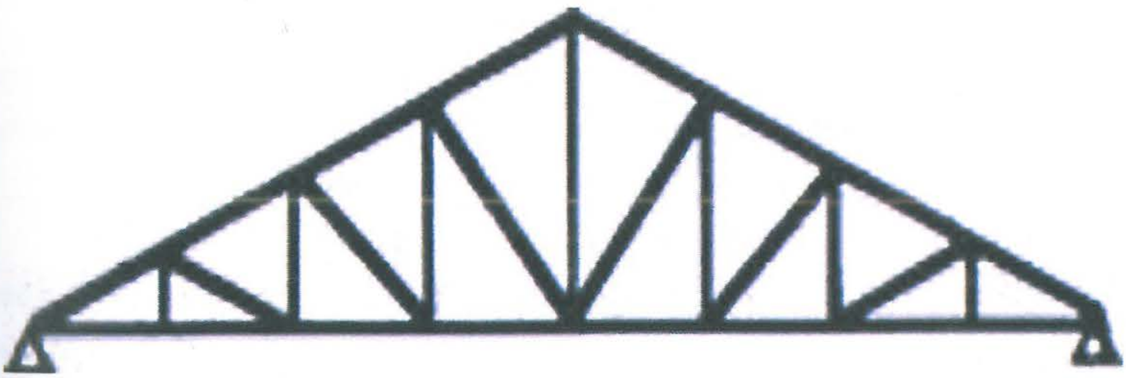

Figure 1.1 Howe truss

Three distinguishing characteristics in the research presented here include: the behaviour of the steel roof trusses determined by the finite element method, the accuracy of the software ANSYS, and the probabilistic analysis for the trusses.

A simple model for the howe shape roof trusses will be proposed and modelled by the software ANSYS in two directions, then finite element calculation were carried out through the software to obtain the deflection, stress and strain of the steel roof trusses, all these analysis were based on finite element method. Probabilistic analysis will be carried out by changing the input parameters to find out which parameters affect the truss effectively. By obtaining the data of the steel roof truss structure, some better suggestions can be made to improve the safety and strength of a steel roof truss system.

1.2 Problem Statement

Traditionally, timber is used for roof truss as a construction material. However, the use of timber is no longer popular due to the increase in cost and is not environmental friendly. Introducing steal as replacement has become modern technique used as construction materials.

Truss is one of the important elements of a structure. Unfit or improper design can lead to the failure of a structure. So, there is a need to provide additional design knowledge and more efficient construction methodologies.

There are several methods can be used to analyzes the behaviour of steel roof trusses. However, among all these methods, finite element method shall be a very effective method to obtain the strength and behaviour of steel roof trusses.

Most of the times, engineers need to find out an accurate data of deflection and stress for the structure before he start design. A lot of time had been wasted for that works. By applying the probabilistic analysis, a range of result can be obtained by input the loading only. So, engineer's works can be reduced and save more time for others.

1.3 Objective

The ultimate purpose of this study is to study the strength and behaviour of steel roof truss by using a finite element analysis software ANSYS.

To achieve this purpose, first step is to determine the result of static structural analysis that can be used to the steel roof truss, which are maximum deflection and von-mises stress of the steel roof truss. Then, probabilistic analysis shall be applied to get the range of the result based on different loading, so the design procedure of an engineer can be simplified.

Finally, the objective of this study is to propose a truss design recommendation for a residential building.

1.4 Scope of Work

First, installs the software of finite element modelling package, ANSYS. Complete all ANSYS tutorials from the website of University of Alberta to learn the skills on doing all structural analysis, either static or dynamic. The tutorials included basic tutorials which outlining basic structural analysis using ANSYS, intermediate tutorials which taught about complex skills such as dynamic analysis and nonlinearities, advanced tutorials which explored the advanced skills such as sub-structuring and optimization.

Then, define the properties of the steel roof truss, which include the dimensions and the sections of the truss. Determine the type of the structure by referring plans or assumptions. Base on the details collected or assumed, model the steel roof truss, the preprocessing state is the steps to define keypoints and lines, geometric data and mesh. It shall be finish to continue to next step.

When the preprocessing state is done, the data is saved since there are few kinds of analysis to be carried out. Loads and supports will be assign and determine the solution of the steel roof truss which are deflection and stress. After that, carry out probabilistic analysis to observe the change of results. Finally, proceed with the thesis writing.

CHAPTER 2

LITERATURE REVIEWS

2.1 Introduction

Nowadays, steel in many structural forms is becoming much more popular and is introduced as an alternative to timber. It is a steelwork technology that has a high potential to be developed in Malaysia. According to Mahmood Md Tahil et al., currently most of the laboratory research and construction practice of cold-formed steel in Malaysia were focus to roof truss system. He suggested that studies on the member capacities of thicker cold-formed sections (range 1.0mm to 3.2mm) can be carried out in order to widen the application area of steel structures, such as wall stub, floor batten, beam column. The cold-formed steel sections can be further applied into sub-structures such as staircases, foundation system, retaining wall and scaffolding [1]. Structural steel generally falls into one of two main categories of hot-rolled steel or cold-formed steel. Cold-formed steel members are formed at room temperature from steel sheet, strip, plates or flat bars in roll-forming machines or by bending operations and have thicknesses typically ranging from 0.4mm to 25mm. Up to 1980's, the thickness was limited to about 3mm. After 1990's, thicknesses 6mm and greater were more commonly produced. Residual stresses exist in hot-rolled steel sections as a result of the nature in which they cool after rolling, whereas residual stresses in light gauge steel result from the cold working process. In general, strain hardening caused by cold-working increases both yield strength and ultimate tensile strength of the steel [2]. The advantages of cold-formed steel are fast erection, lightweight, clean and easier construction. It had become a brilliant building component in the construction industry. However, the most common and useful steel structure in Malaysia is hot roll steel. Since hot roll steel is more functional in Malaysia, this research is going to analyzes roof truss built by using common steel, which is hot roll C-channel steel.

From Hancock's journal, he review includes 50 papers published in the Journal of Constructional Steel Research, Thin-Walled Structures and the Journal of Structural Engineering, American Society of Civil Engineers on cold-formed steel research and design. They have been grouped into the major areas of:

1. Compression members
2. Distortional and element buckling
3. Corrugated and curved panels
4. Flexural members and purlins
5. Torsional and distortion
6. Web crippling
7. Connections and fasteners
8. Mechanical properties
9. Composite and plasterboard construction
10. Storage racks
11. Optimisation

He also introduce about a method developed by Schafer and Pekoz called the "Direct Strength Method" which may simplified design of cold-formed steel structural members [3]. In this research, all major area mentioned in Hancock's paper shall be consider and analyse to understand the strength of steel structure clearly. However, the method use to analyse will be the numerical method named "finite element method", but not "Direct Strength Method". The main difference between these two methods is that finite element doesn't simplify the design of steel structural members. To do the analysis, at first a simple design of steel roof truss is apply, as a simple design structure is easy to control and analyse, thus finite element analysis will be a suitable and appropriate in analysing this structure.

2.2 Steel Roof Truss

Steel roof trusses are frequently constructed with C-section or Z-section as chord as shown in figure 1 and are designated as either in-line or offset.

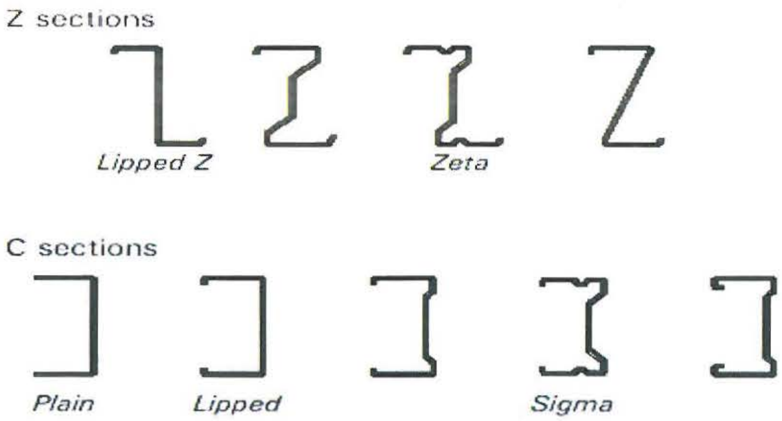

Figure 2.1: Common shape for steel

Steel roof trusses commonly use C-sections as chord and web members and are designated as either in-line or offset. An in-line truss is one in which both web and chord members are kept in a single plane. This is achieved by copying the flanges of the C-sections at certain locations allowing the webs of the members to fit together [4]. The term "offset" encompasses every truss in which all members do not lie within a single plane. These trusses typically have member webs connected back-to-back with one of two cases:

(1) Top and bottom chords are laid in-line and webs are offset;
(2) Top chords are laid in-line with webs and bottom chords are offset.

At the support where the highest loads occur, offsetting the top and bottom chords from one another at this location could help improve the stability of the heel connection because of opposing tendencies of the members to rotate. On the other hand, the offset configuration may be expected to cause warping and torsion stresses in chords due to the eccentricity of the loading, thus sacrificing some loading capacity [5]. However, from an economic viewpoint, offset fabrication of trusses can reduce both labour and plate material costs [6]. In this research, steel roof trusses with C-section will be modelled and tested by using finite element method, while the member webs connection of trusses will apply the steel roof trusses with first case mentioned in above, which is top and bottom chords are laid in-line and webs are offset.

In general, steel roof trusses are currently manufactured with proprietary sections or commonly available C-sections as chord and web members. Owing to an overall uncertainty in the industry especially with regard to strength and performance, member connections are generally over-designed and therefore somewhat inefficient. A literature review yielded limited research findings in both experimental testing and numerical analysis on cold-formed steel roof trusses and their connections.

Harper studied the behaviour of top chords by testing full-scale Fink trusses with a span of 6 m and recommended that top chords be designed as concentrically loaded compression members [7]. Riemann conducted full-scale tests and computer analyses to investigate the capacity of compression webs and suggested that they may be designed as beam–columns [8]. LaBoube and Yu conducted two experimental studies to evaluate compression webs and top chords of offset trusses. The web crippling failure at loaded panel points was the predominant failure mode for the top chord. An interaction equation for the design of top chords was proposed where the combined action of axial compression, bending moment, and web crippling was recognized [9]. From their research, all of their analysis applied by using either experimental method or full scale tests, so this research will shows a different method which is finite element analysis to solve all the problems by using ANSYS.

Wood tested 23 small-scale and 10 full-scale offset CFS roof trusses. Heel plates were used to reinforce the connection between top and bottom chords at heel location and to accommodate the height required by a raised heel construction. It was found that the heel plate had a significant influence on the strength and that local buckling of top chords adjacent to heel plates was the primary failure mode. The effectiveness of configurations and thicknesses of the heel plate were also investigated [10]. This test also can be done in a different way by using finite element method.

2.3 Finite Element Analysis

In the research of K.S. Sivakumaran, a finite element analysis model for the post-local buckling behaviour of steel members subjected to axial compression has been developed. The finite element model consists of a Total Lagrangian nonlinear 9-node "assumed strain" shell finite elements, and experimental-based material properties models to represent the body of the steel sections. Experimentally derived residual stress variations, and initial geometric imperfections have also been incorporated. A special loading technique and a displacement solution algorithm were employed to obtain a uniform displacement condition at the loading edges. Details of a test program involving 20 non-perforated, and perforated cold-formed stub-column steel sections have been presented in the second part of the paper. The comparison between the test results, and the finite element results was performed for axial and lateral displacement behaviour, buckling loads, ultimate loads, and axial stress distribution. The comparison forms the basis for the evaluation of the efficiency, and the accuracy of the finite element model, and it indicated that the finite element analysis model constructed herein gives accurate and consistent results for the behaviour of the cold-formed steel members subjected to axial compression. The model was also capable of predicting the pre-buckling stress concentrations at locations adjacent to the perforation. Perforated stub-column specimen showed lower axial stiffness, lower buckling and ultimate loads, than the non-perforated specimen [11]. It showed that finite element analysis is available and suitable to be use as the method to obtain the behaviour of steel accurately and efficiency.

According to Ronghui Wang's research, mechanical behaviours of a structure could be conducted by model test and numerical finite element analysis. The model test shows that the maximum stress in the joint is less than the material allowable stress and the maximal stress induced from the secondary moment accounts for about 30% of the total stress. A three-dimensional finite element model is used in the numerical analysis and the results are in very good agreement with those of the model test. The two sets of results have similar distribution patterns, and ratios between the experimental results to numerical analysis results are mostly within the range from 0.8 to 1.2, and the relative errors for most of the results are no more than 10% [12]. This means that, for the design of a complex joint, finite element analysis is a reliable method for predicting the complex stress distribution in the structure. As the finite element analysis is far more cost effective than the model test approach, the promotion of its application in design is worthwhile.

In Dan Dubina's research, four complex problems of analysis of structures is solved using testing, which are:

(1) Calibration of imperfection factor for interactive local-overall buckling of cold-formed steel compression members. On this purpose, test results collected from literature, ECBL approach, and statistical data processing procedure of Annex Z from ENV1993 (actually, replaced by Annex D of EN1990) have been used.
(2) Experimental calibration of stiffness of joints in cold-formed steel trusses. Tests on single-bolt lap joints and on structural T-joints have been used to calibrate the stiffness calculation formula. Afterwards, a full-scale test on a truss module has been used to validate the proposal.
(3) Analysis and design of cold-formed steel pitched-roof portal frames. First, a test campaign on ridge and knee joints was performed in order to characterize their behaviour and performance. Based on these test results, the component method from EN1993-1-8 has been adopted for cold-formed steel joints calculation in terms of stiffness and strength. In the second phase, full-scale tests on frame units have been used to evaluate and confirm the global analysis models.
(4) Analysis and design of cold-formed steel framing houses. Tests on shear panels and fastening systems combined with numerical simulations have been carried out in order to propose a design approach based on the shear walls capacity and 3D analysis. Next, *in situ* vibration measurements on a building structure, in subsequent stages of erection, have been used to evaluate the contribution of finishing and to validate the design approach.

In all these four problems, tests only or accompanied by numerical simulations have been used to find the proper solutions and/or to validate them. In fact, without tests, the solutions of these problems would be very difficult, say impossible, to be found [13]. From Dan Dubina's review paper, noted that she had used few kind of method in determining the data of cold-formed steel. So, in my research, I will try to determine some of the behaviours of a steel truss by only apply finite element analysis, which include the deflection, maximum stress and strain of the steel truss

There is a review paper written by A. Zafer Senalp mentioned about study static dynamic and fatigue behaviour of newly stem shapes for hip prosthesis using finite element analysis carry out by ANSYS [14]. His study is almost same as this research since this research also study about a structure by using finite element analysis, and also carry out by ANSYS. His study and methods are suitable to be referred since there are so many similarities. From his study, he built 3 finite element models that required in finite element analysis by discretizing the geometric models into smaller and simpler elements. The discretization was performed in ANSYS environment. Compare with this study, a finite element model will be created for finite element analysis but without discretizing the model. The modelling of steel roof trusses is performed by ANSYS too.

In the literature, structures are often designed according to the results of static analysis. Static finite element analysis are mostly conducted under body weight loads but since trusses doesn't have any impose load, so only self-weight of the trusses and wind load will be consider. However, dynamic effects may add up to about 10-20% or more loading to the structure which must be taken into account not to cause fracture or fatigue failure of the steel roof trusses.

The other journal which written by Asokendu Samanta is finite element static and dynamic analysis of folded plates which introduce on finite element formulation of corrugated sheet [15]. The stiffness matrix mentioned in his study will be use in this research, which was:

$[K_e] = \int\int [T]^T[B]^T[D][B][T]\, d_x d_y$

Where is the transformation matrix which correlates the midside nodal degrees of freedom to the corner nodal degrees of freedom, [B] is the strains to the nodal displacements and [D] is the rigidity matrix. Integration is performed by using three point Gauss quadrature rule in area coordinates. To determine the element stiffness matrix in local coordinates of a particular element the global coordinate of each node of that element is transformed into the local coordinate. The stiffness matrix in local coordinates is then transformed into corresponding global coordinates. A general shell surface is generated using this faceted element. After the generation of element stiffness matrix and assembly, the final solution follows the standard procedure. Further more, he also mentioned about the formulation of nonlinear static finite element, fatigue analysis, free vibration analysis, etc. However, this study is aim to understand the effectiveness of each input variables on affecting the strength of the steel truss, so only static analysis will be carried out.

2.4 Probabilistic Analysis

A journal written by Stefan Reh had introduced about probabilistic finite element analysis by using ANSYS [16]. In his journal, he wishes to solve the problems or challenges under the existence of randomness and uncertainty, thus he apply the software ANSYS by running probabilistic finite element analysis to solve various industrial example problems. He also mentioned about some advantages and disadvantages of both software ANSYS and probabilistic analysis. This study is similar with his journal which applied a steel roof truss and analyzes it using ANSYS and finite element analysis analysis. Similarly, this study also uses Monte Carlo simulation method as the method on doing probabilistic analysis. Based from his journal, he can proved that ANSYS can provide a proper and accurate results for both finite element analysis and probabilistic analysis.

Another journal written by Hyuck Jin Park also applied probabilistic analysis to solve the uncertainty of input data and analysis results [17]. Although he solve the problem of engineering geology, the concept of probabilistic analysis shall be the same. Similar with this research, he apply Monte Carlo method to carry out probabilistic analysis. In his journal, he stated out many random input data and find the results from his study area.

CHAPTER 3

METHODOLOGY

3.1 Introduction

In this present study, a steel roof truss will be model and analyse from 2 sides to obtain the results. Here, a model will be prepared with an assumed size by using ANSYS software. The basic specifications and properties of steel are showed in table 3.1.

Table 3.1: Specifications and Properties of Steel Roof Truss

Properties	Value	Unit
Density	7850	Kg/m³
Elastic modulus	210	GPa
Shear modulus	77.9	GPa
Poisson's ratio	0.3	
Cross Section Area	1000	mm²
Shape	Howe	

The ANSYS analysis consists of 3 phases, which are:

a) Preprocessing phase- defining the problem, model the structure
b) Solution phase- Assigning the loads and constraints and solve the resulting system of equations.
c) Postprocessing phase- Viewing the results